An Overset Mesh Methodology for CFD Modelling of Rotary Compressors

Wai Chang Poh
Nanyang Technological University, Singapore, waichang001@e.ntu.edu.sg

Kim Tiow Ooi

Basman Elhadidi

An Overset Mesh Methodology for CFD Modelling of Rotary Compressors

Wai Chang POH[1], Kim Tiow OOI[2], Elhadidi BASMAN[3]

[1,2,3]School of Mechanical and Aerospace Engineering,
Nanyang Technological University, Singapore
[1]WAICHANG001@e.ntu.edu.sg
[2]MKTOOI@ntu.edu.sg
[3]MBASMAN@ntu.edu.sg

ABSTRACT

Full 2D and 3D unsteady CFD simulations are gradually replacing the widely used lumped parameters simulations for rotary compressors. Lumped parameter models predict the overall thermodynamic processes; however, they cannot reveal the spatial and temporal variations in the working chamber and account for complex fluid interactions and losses. The complex and continuously deforming working chamber geometry is challenging for CFD. The overset method has been implemented here for the simulation of a 2D Coupled Vane Compressor (CVC). The method was compared to a dynamic mesh formulation using a remeshing technique and has shown; (1) it is 3-4 times faster computationally and (2) capable of modelling tight clearances not possible in the dynamic mesh approach and are difficult, if not impossible to model using the lumped parameter models. The CFD results revealed complex fluid interactions inside the chamber, accounted for leakage and can be used to optimize the CVC designs further.

1. INTRODUCTION

Historically, the lumped parameters approach was used for the performance prediction of rotary compressors. The recent advancements of computer hardware and software propelled the research of rotary compressors towards 2D or 3D CFD simulation studies. CFD modelling reveals the hidden information in the working chamber of the compressor and leads to a much better understanding of the flow physics. However, the primary challenge of CFD models is to generate the complex solution domain of the working chamber, especially for rotary compressors with its varying geometries during operation. Several meshing approaches for rolling piston compressor (Ding & Gao, 2014), screw compressor (Kovačević, 2005), and sliding vane compressor (Bianchi *et al.*, 2017) are available in the literature. However, these approaches can only cater to the relatively simple geometry deformations.

In this paper, the Coupled Vane Compressor (CVC), which was developed by Ooi & Shakya (2018) is used to showcase the complex geometry changes due to its unique dual vane configuration. Figure 1 illustrates the working principle of the CVC with the highlighted areas showing the working chamber deformations in a complete operating cycle of 540°.

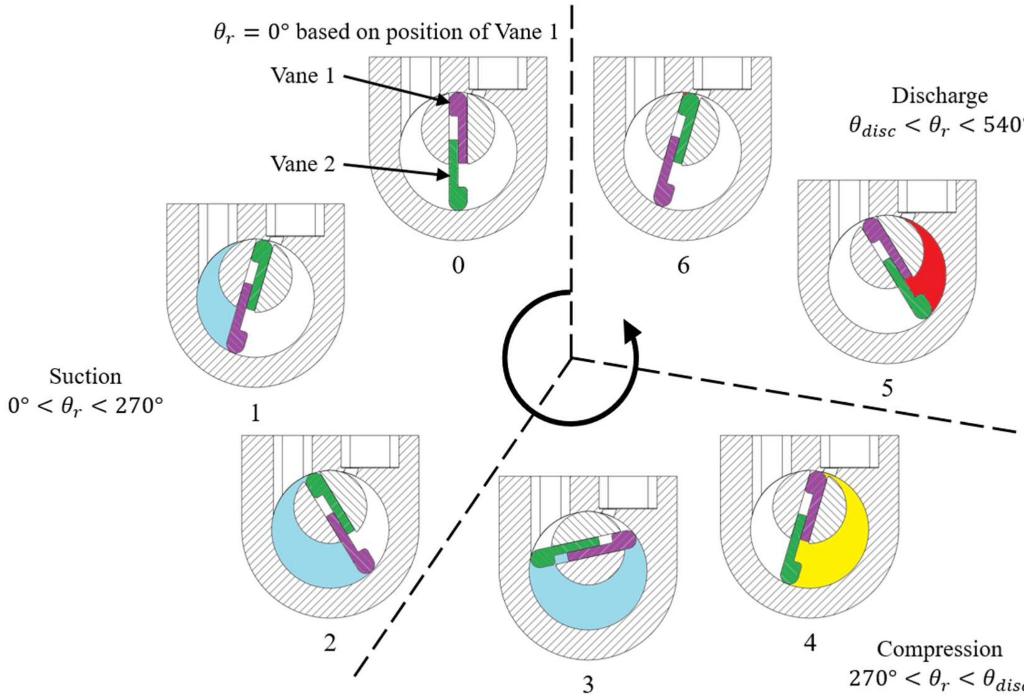


Figure 1: Operating cycle of CVC

2. GEOMETRY AND MESHING STRATEGIES

2.1 Geometry

The 2D geometry is taken at the midplane of CVC, where it encompasses the suction and discharge ports. Since the first objective is to investigate the meshing strategy, Geometry A is used as the model for the dynamic mesh and overset mesh comparison study, as shown in Figure 2. Geometry A is simplified by removing the discharge chamber and the complexity of the discharge valve from Geometry B. Furthermore, a larger gap, Gap A is introduced between the vanes to reduce the computational time by relaxing the calculations in the chamber formed by the vanes. Besides that, the feasibility of the meshing strategies used for CVC can be determined since the solution domain must remain continuous, especially at the initial clearances at the sliding regions of the two components of 100 μm, as shown in Figure 3 and Figure 5. From the initial investigations of utilizing 100 μm clearances, it was discovered that there are significant internal leakages, and the compression process could not achieve the desired discharge pressure. Thus, Geometry B is used onwards after the initial investigations comparing the meshing approaches using 50 μm clearances.

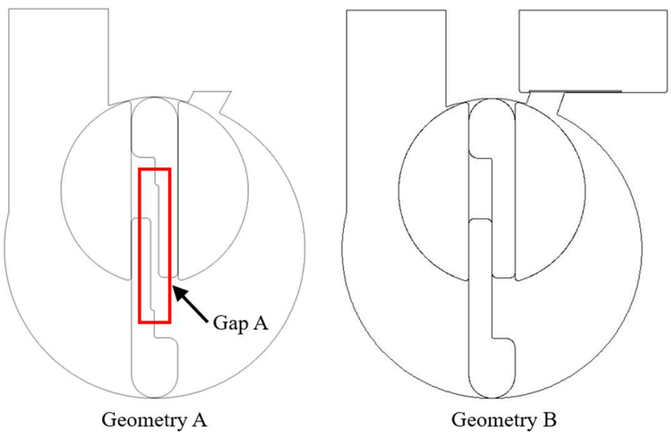


Figure 2: Geometry of the solution domains for CVC

2.2 Dynamic Mesh with Remeshing

As the remeshing feature using the dynamic mesh approach for the moving and deforming solution domain can only cater to triangular elements in the case of ANSYS computational code, the mesh is generated in most of the solution domain using quadrilateral elements. At the interface of the sliding regions, the mesh was generated using triangular elements, as shown in Figure 3. The two zones are connected using a non-conformal interface. When the elements within the deforming zone deformed due to the motion of the rotor and vanes, the solver generates new elements that conform to the specified mesh quality, which is known as the remeshing process. The limitation of this approach is the resulting large number of elements since the smallest element size at the clearance dictates the overall size of all the elements in the deforming region.

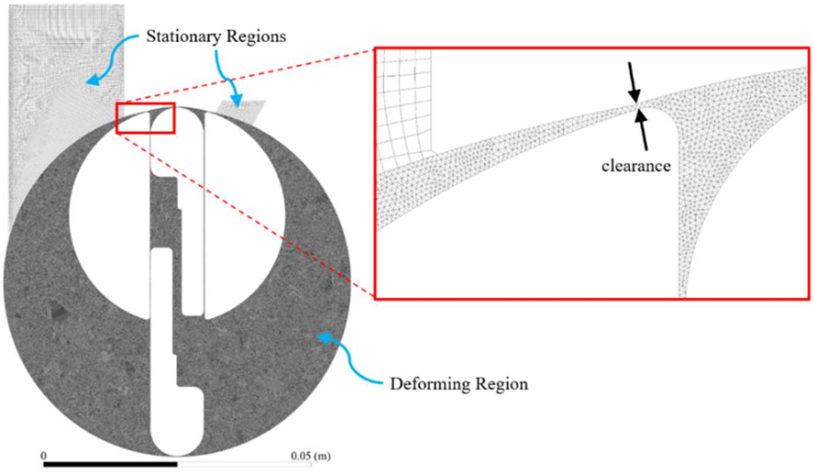


Figure 3: Mesh of Geometry A for dynamic mesh with remeshing

2.3 Overset Mesh

The overset mesh approach has been utilized successfully in other research areas with external flow systems such as ocean engineering (Hao *et al.*, 2019) and aerospace with guidelines provided by Chan *et al.* (2002). In the case of internal flow systems, Suman *et al.* (2016) attempted to model a single screw expander using such method, but further investigations are required to model the clearances and aid solution convergence for transient simulations. The overset mesh approach consists of several overlapping meshes to form the solution domain that consists of active elements to be solved. Meanwhile, the other elements (passive elements) which are not within the solution domain are turned off temporarily or permanently depending on the motion of each component. The active elements consist of donor elements that transfer information from a mesh to the receptor elements on the other mesh at the overset interface.

The individual component meshes are prepared separately with quadrilateral elements, as shown in Figure 4. Although the meshing procedure seems reasonably straightforward, special attention must be given to the boundary layer elements and the element size at the overset boundaries for each component. The boundary layer elements at the clearances of 50 μm for Geometry B is shown in Figure 5, consisting of several elements across it which can be further refined by the mesh controls. Hence, the flow details at the clearances can be captured more accurately as compared to a single cell thick element used by Ding & Gao (2014). Besides that, the overset mesh also requires a minimum number of overlapping elements at the overset boundary, especially between the near-wall regions at the clearances and to prevent orphan cells.

At the overset interfaces, the element size and mesh continuity are vital to ensure the accuracy of the solution. A 1:1 ratio of the element size at the overset interface is preferred to minimize interpolation errors from the different meshes, as shown in Figure 5. Figure 6 illustrates the case where the vane mesh is discontinuous from the vane wall and caused an anomaly in the pressure contour. This is because the priority is given to the smaller element size at the overset interface. Hence, the problem can be resolved by implementing a banded region of smaller element size in the background mesh as well as removing the outer layer of vane mesh at the vane tip, as shown in Figure 4. Similar modification is also performed at the discharge port area. Remeshing is not required since the elements do not deform during the motion of the components. Thus, the mesh quality generated initially is preserved. Figure 7 illustrates the solution domains which are formed by overlapping each meshed component onto the meshed background.

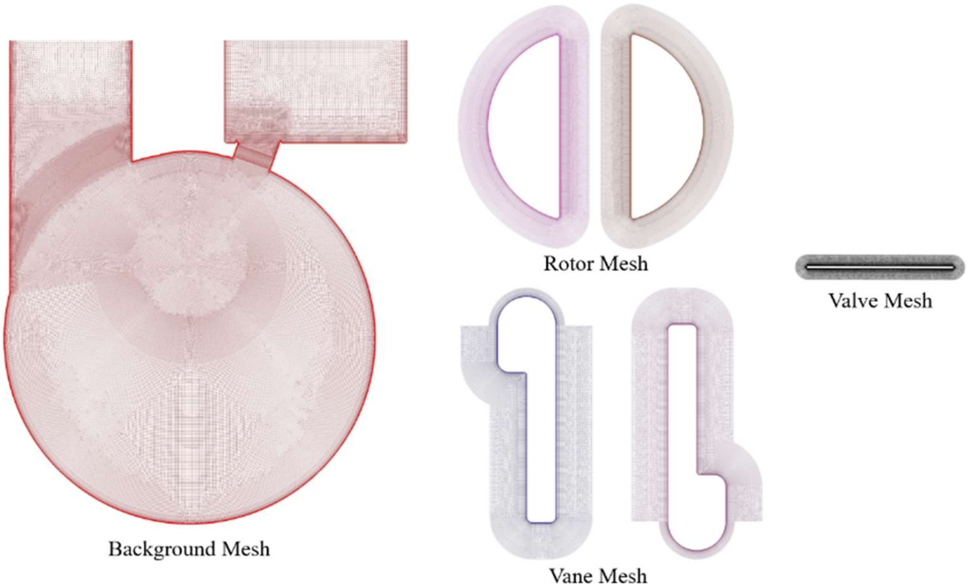


Figure 4: Component meshes for overset mesh

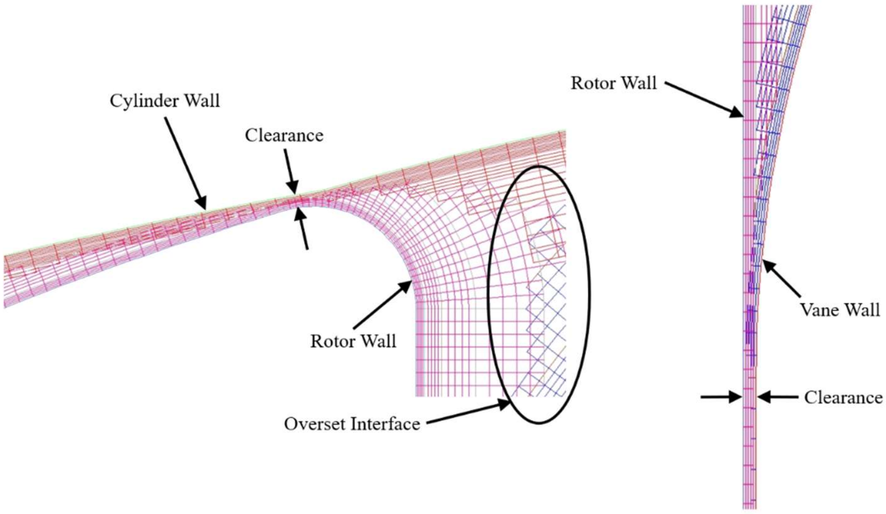


Figure 5: Elements at the clearances for overset mesh

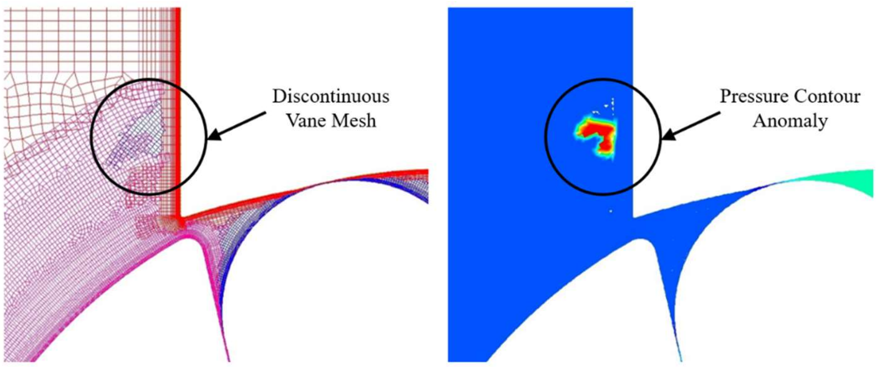


Figure 6: Pressure contour anomaly from discontinuous vane mesh

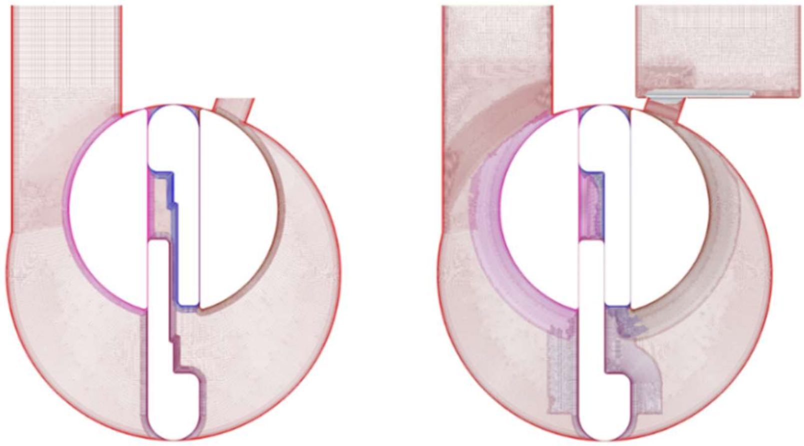


Figure 7: Generated solution domain from overset mesh

3. SIMULATION SETUP

The 2D transient adiabatic CFD simulations are carried out using the commercially available code ANSYS Fluent 2019 R3 employing the realizable $k-\epsilon$ turbulence model from the fully turbulent nature of the flow in CVC and provides a relatively accurate solution with a faster solution convergence rate as compared to the $k-\omega$ turbulence model. Air is used as the working fluid and modelled as an ideal gas. Table 1 lists the boundary conditions with the operating speed of 1500 rpm. Meanwhile, the displacements of the moving parts are controlled using user-defined equation of motions. The valve modelling is simplified as an equivalent 1D torsional spring model derived from the Euler-Bernoulli beam theory (Ding & Gao, 2014). It is defined with the six degrees of freedom motion where the valve is displaced based on the resulting fluid forces acting on it in a one-way fluid-structure interaction.

Table 1: CFD boundary conditions

	Suction Port	**Discharge Port**
Type	Pressure Inlet	Pressure Outlet
Pressure (Pa)	101,325	506,625
Temperature (K)	300	475

4. RESULTS AND DISCUSSIONS

4.1 Dynamic Mesh with Remeshing and Overset Mesh Comparison – Geometry A

Geometry A, which consists of 100 µm clearances, is used for the comparison study utilizing both the meshing approaches. The discharge port pressure indicated that the compression of the working fluid failed to achieve the specified discharge pressure at 1500 rpm. So, another set of simulation was performed at 3000 rpm, and only the overset mesh could achieve the targeted discharge pressure, as illustrated by Figure 8. The discrepancy between the pressure curves is caused by the mesh quality from the two meshing approaches. Remeshing was applied in the dynamic mesh method, which deteriorates the mesh quality as the simulation progresses, especially at the clearances. However, the overset mesh does not require any remeshing and the mesh quality is preserved throughout the simulation.

The effects of internal leakages due to the 100 µm clearances and Gap A are also perceived from the discharge port pressures that could not achieve the specified discharge pressure. Although Geometry A highlighted the issue of internal leakages, it has shown that the dynamic mesh with remeshing produced undesirable results due to the degrading mesh quality from remeshing. It is shown by the decreasing peak pressure caused by the accumulated interpolation errors from the previous mesh to the newly generated mesh. Table 2 summarizes the computational cost where the overset mesh is 3 to 4 times more cost-efficient as compared to the dynamic mesh with remeshing. Thus, Geometry B with 50 µm clearances utilizing the overset mesh is employed for the following study.

Table 2: Computational cost between dynamic mesh and overset mesh

	Dynamic Mesh	Overset Mesh
Number of nodes	218,701	195,929
Number of elements	422,826	191,126
Computational time per rotor revolution (32 cores)	36 – 48 hours	11.6 – 12.8 hours

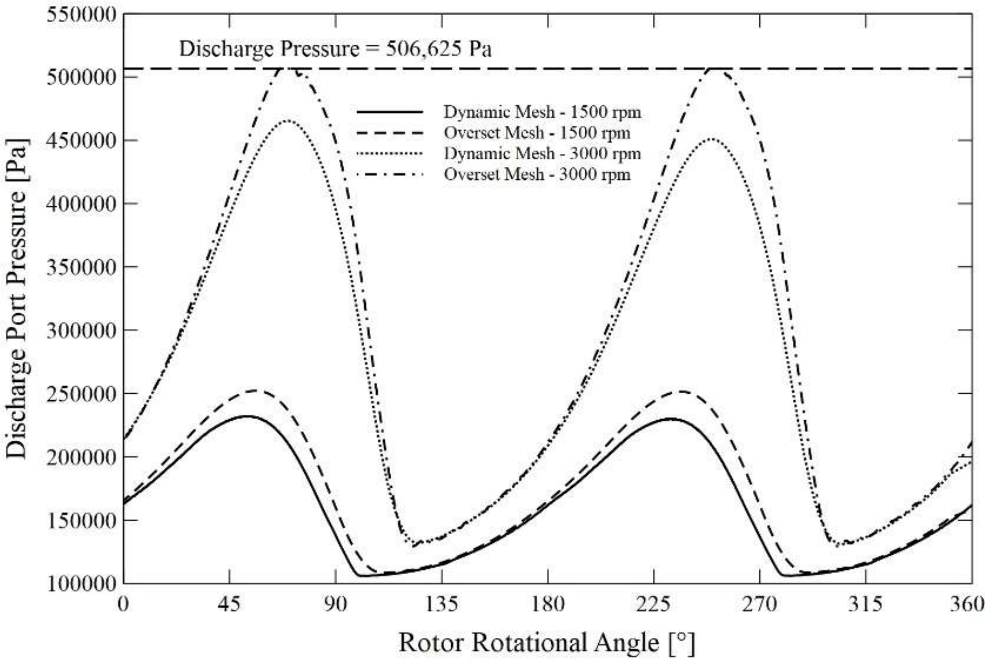


Figure 8: Variation of discharge port pressure for Geometry A

4.2 Overset Mesh Results – Geometry B

Figure 9 illustrates the variation of working chamber pressure obtained from the 2D transient CFD simulation for Geometry B utilizing the overset mesh approach. The variation of working chamber volume is used as the indicator for the processes in one operating cycle. Pressure probes were placed within the working chamber and the average working chamber pressures were obtained for each rotor rotational angle. The averaged working chamber pressures are then assembled to obtain the working chamber pressure curve. The suction process occurs as the volume of the working chamber rises to its maximum. The compression process begins at 270° rotor rotational angle until the discharge process starts at 391°. The lumped parameters model from Shakya & Ooi (2020) showed similar compression and discharge starting points.

Although there are over-compression effects due to the valve stiffness, experimental studies are required to validate the accuracy of the valve models. Conversely, the CFD result indicates the presence of internal leakage at 454° due to the vane tip passes the discharge port where the undischarged compressed working fluid in the discharge port leaks to the next cycle. However, the lumped parameters model assumed that all the working fluid would continue to be compressed and discharged after that point.

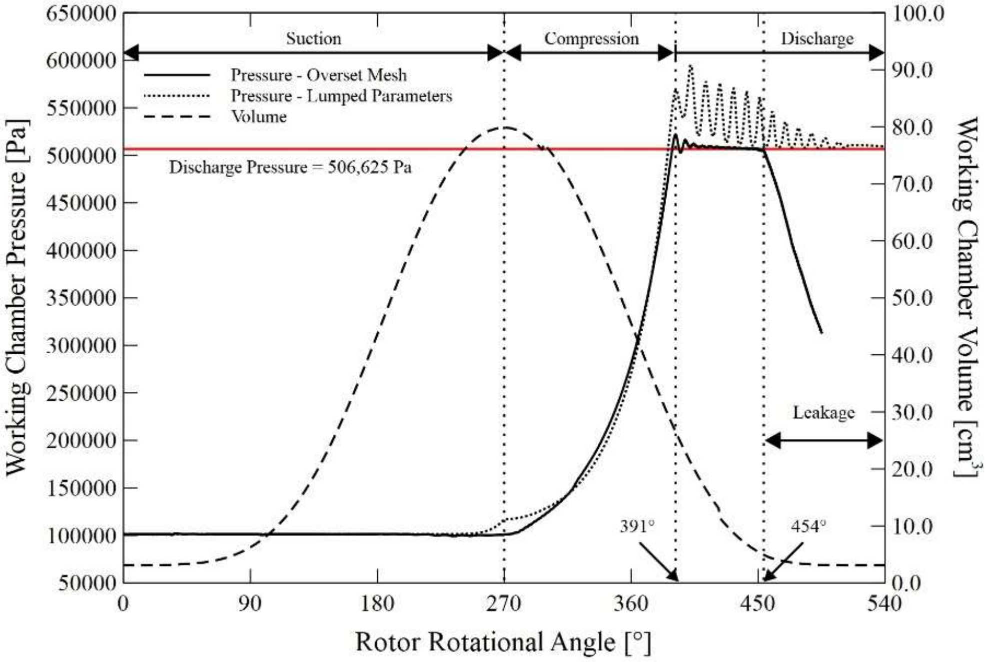


Figure 9: Variation of working chamber pressure and volume

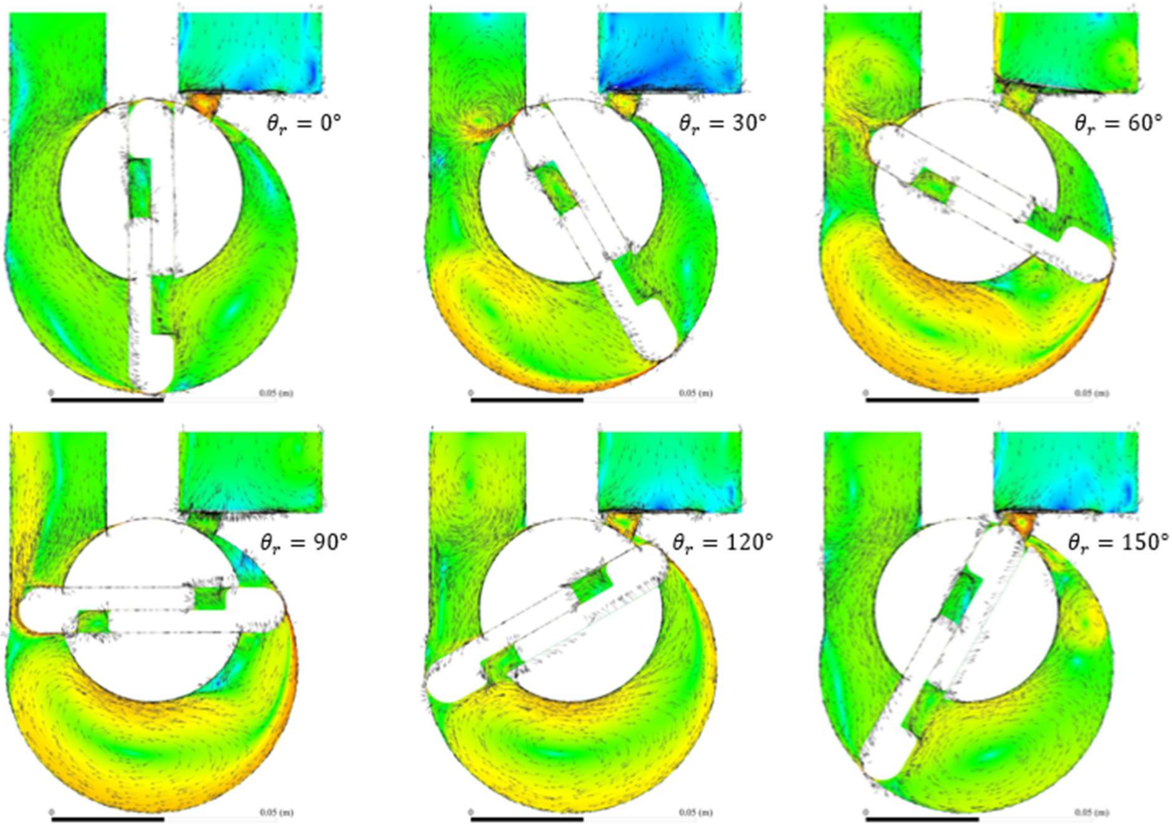
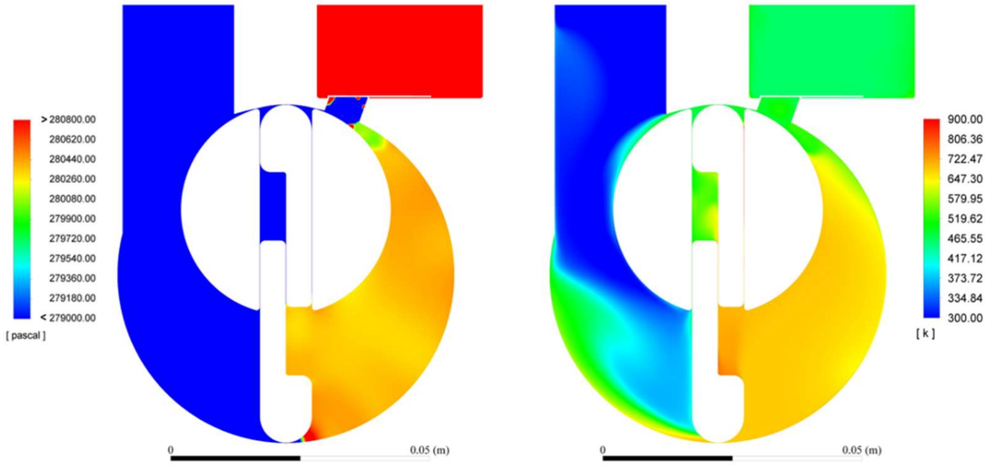


Figure 10: Velocity flow field of CVC



Pressure Contour Temperature Contour

Figure 11: Pressure and temperature contours spatial variations in the working chamber

Figure 10 illustrates the snapshots of the velocity flow field for every 30° rotor rotational angle. The figure reveals the presence of circulation zones due to the effects of internal leakages. Furthermore, the spatial variation of pressure and temperature in the working chamber is shown in Figure 11, where the information can be analyzed further to enhance the design and hence the performance of CVC which is currently underway.

5. CONCLUSIONS

The overset mesh method has been applied to simulate the two-dimensionally transient working processes of the Coupled Vane Compressor (CVC) and compared to that using the dynamic mesh with remeshing method. The results show that the overset meshing approach is capable of modelling tight clearances at the rubbing regions while preserving the mesh quality throughout the entire simulation process without the need to regenerate new meshes as the solution domain deforms during a complete compressor cycle. It was also noted that the computational cost was 3 to 4 times cheaper than that of the dynamic mesh approach. Furthermore, the results were validated by the lumped parameters approach with good agreement. It is concluded that the overset mesh method is superior, as compared to the dynamic meshing approach with mesh regeneration, in modelling deforming and moving boundary problems such as those found in positive displacement compressors. It is believed that this overset mesh approach is very useful and practical to be applied in improving the design of the compressor. Obviously, the 3D transient model will be able to provide an even more realistic prediction which accounts for all the flow interactions caused by the end effects.

NOMENCLATURE

θ	rotational angle	(°)

Subscript

r	rotor
disc	discharge

REFERENCES

Bianchi, G., Rane, S., Kovačević, A., & Cipollone, R. (2017). Deforming grid generation for numerical simulations of fluid dynamics in sliding vane rotary machines. *Advances in Engineering Software, 112*, 180-191.

Chan, W. M., Gomez, R. J., Rogers, S. E., & Buning, P. G. (2002). Best practices in overset grid generation. *32nd AIAA Fluid Dynamics Conference and Exhibit*, (p. 3191).

Ding, H., & Gao, H. (2014). 3-D Transient CFD Model For A Rolling Piston Compressor With A Dynamic Reed Valve. *International Compressor Engineering Conference.*

Hao, C., Ling, Q., Zhihua, M., Wei, B., Ye, L., Derek, C., & Clive, M. (2019). Application of an overset mesh based numerical wave tank for modelling realistic free-surface hydrodynamic problems. *Ocean Engineering, 176*, 97-117.

Kovačević, A. (2005). Boundary adaptation in grid generation for CFD analysis of screw compressors. *International Journal for Numerical Methods in Engineering, 64*(3), 401-426.

Ooi, K. T., & Shakya, P. (2018). A New Compact Rotary Compressor: Coupled Vane compressor. *International Compressor Engineering Conference.*

Shakya, P., & Ooi, K. T. (2020). Introduction to Coupled Vane compressor: Mathematical modelling with validation. *International Journal of Refrigeration, 117*, 23-32.

Suman, A., Ziviani, D., Gabrielloni, J., Pinelli, M., De Paepe, M., & Van Den Broek, M. (2016). Different Numerical Approaches for the Analysis of a Single Screw Expander. *Energy Procedia, 101*, 750-757.

ACKNOWLEDGEMENT

The authors would like to thank Nanyang Technological University, Singapore for the opportunity and support to carry out the study.